PARAMETRIC STUDY ON FATIGUE FAILURE USING CRACK INITIATION METHOD

MOHD FADZRUL HISYAM BIN BADARUDIN

Report submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

ABSTRACT

This thesis presents the parametric study on fatigue failure using crack initiation method analysis. In order to investigate the fatigue failure of AISI/SAE 1045 steel by varying the microstrain parameters under different loading cases and predict the crack initiation life, compact tension specimen model is used for this project. The finite element linear static analysis is done by using MSC Patran 2010 and MSC Nastran 2007 to determine the result of the stress distribution on the model. The fatigue analysis is conducted by using ANSYS 13.0 nCode DesignLife to determine the microstrain value that initiates the dynamic and static failure under three loading cases; compression-compression, compression-tension and tension-tension. The obtained microstrain values are then used to obtain the fatigue damage, D which is then used to predict the crack initiation life of the material. Using Palmgren-Milner rule, the value of D will obtain the crack initiation life cycle of the material, N_f. Based on the result, it has found that the most critical loading case for the fatigue and static failure are the tension-tension. The crack initiation life of the material is at high cycle fatigue, and able to stand about 5×10^6 cycles without failure for every loading case. Meanwhile the static failure analysis has found that there is no crack initiation life, as it has failed at very low cycle in every loading case.

ABSTRAK

Tesis ini membentangkan kajian kepelbagaian parameter terhadap kegagalan lesu dengan menggunakan prosedur analisis permulaan rekahan. Untuk mengkaji kegagalan lesu keluli AISI/SAE 1045 dengan mempelbagaikan parameter regangan mikro di bawah kes-kes pembebanan yang berbeza dan meramal jangka hayat permulaan keretakan, model spesimen tegangan padat digunakan untuk projek ini. Analisis statik mendatar unsur terhingga dilakukan dengan menggunakan perisian MSC Patran 2010 dan MSC Nastran 2007 untuk mendapatkan keputusan pengagihan tekanan terhadap model. Analisis kelesuan dijalankan menggunakan perisian ANSYS 13.0 nCode DesignLife untuk mendapatkan nilai regangan mikro yang memulakan kegagalan dinamik dan statik di bawah tiga kes bebanan; kes tekanan-tekanan, tekanan-tegangan, dan tegangan-tegangan. Dari nilai regangan mikro yang diperoleh, kemudian digunakan untuk mendapatkan nilai kerosakan lesu, D yang digunakan untuk meramal jangka hayat permulaan keretakan bahan tersebut. Dengan menggunakan hukum Palmgren-Milner, nilai D akan mendapatkan kitaran jangka hayat bahan tersebut, N_f. Dari hasil yang diperoleh, difahamkan bahawa kes bebanan yang paling kritikal untuk kegagalan lesu dan statik adalah kes bebanan tegangan-tegangan. Jangka hayat permulaan keretakan untuk bahan tersebut adalah pada kitaran lesu tinggi, dan mampu bertahan kira-kira 5×10^6 kali kitaran untuk setiap kes bebanan. Sementara itu, analisis kegagalan statik mendapati bahawa tiada permulaan keretakan berlaku pada bahan tersebut, memandangkan bahan tersebut telah gagal pada kitaran yang sangat rendah, untuk setiap kes bebanan.

TABLE OF CONTENTS

		Pages
EXAMINER'S DECLARATION		ii
SUPERVISOR'S DECLARATION		iii
STUDENT'S DECLARATION		iv
DEDICATION		v
ACKNOWLEDGEMENTS		vi
ABSTRACT		vii
ABSTRAK		viii
TABLE OF CONTENTS		ix
LIST OF TABLES		xi
LIST OF FIGURES		xii
LIST OF SYMBOLS		xvii
LIST OF ABBREVIATIONS		xviii
CHAPTER 1	**INTRODUCTION**	
1.1	Project Background	1
1.2	Problem Statement	2
1.3	Project Objectives	3
1.4	Project Scopes	3
CHAPTER 2	**LITERATURE REVIEW**	
2.1	Introduction	4
2.2	AISI/SAE 1045 Steel	5
	2.2.1 Steel	5
	2.2.2 Carbon Steel Classifications	5
	2.2.3 AISI/SAE 1045 Steel	6
2.3	Introduction of Metal Fatigue Failure Analysis	7
	2.3.1 Types of Fatigue Failure Analysis	7
	2.3.2 Finite Element Analysis (FEA)	8
	2.3.2.1 Pre-processing	9
	2.3.2.2 Analysis	9

		2.3.2.3 Post-processing	9
2.4	Crack Initiation		10
	2.4.1	Fatigue	10
	2.4.2	Crack Initiation Life	10
2.5	Specimen Geometry		11
	2.5.1	Standard Test Specimen	11
	2.5.2	Compact Tension Specimen	11
2.6	Summary		13

CHAPTER 3 METHODOLOGY

3.1	Introduction		14
3.2	Flow Chart		14
3.3	Methodology Steps		16
	3.3.1	Linear Static Analysis	27
	3.3.2	Fatigue Failure Analysis	25

CHAPTER 4 RESULTS AND DISCUSSIONS

4.1	Introduction		33
4.2	Finite Element Analysis Results		33
	4.2.1	Input Parameters	34
	4.2.2	Crack Initiation Analysis Results	34
4.3	The Theoretical Failure Stress		72
4.4	Crack Initiation Life		74

CHAPTER 5 CONCLUSSION AND RECOMMENDATIONS

5.1	Conclusions	76
5.2	Recommendations	77

REFERENCES 78

LIST OF TABLES

Table No.	Title	Page
2.1	AISI/SAE 1045 steel element composition.	6
2.2	AISI/SAE 1045 steel mechanical properties	7
3.1	Material properties of the specimen	21
3.2	The variants of mesh seed sizes	22
3.3	Input parameter for crack initiation studies	27
4.1	Input parameter for dynamic and static analysis studies	34
4.2	Dynamic failure initiation microstrain and damage value	71
4.3	Static failure initiation microstrain and damage value	72
4.4	Theoretical failure stress at dynamic failure	73
4.5	Theoretical failure stress at static failure	73
4.6	Crack initiation life cycles for dynamic failure analysis	74
4.7	Crack initiation life cycles for static failure analysis	75

LIST OF FIGURES

Figure No.	Title	Page
2.1	CT specimen dimension	12
3.1	The flowchart of the project methodology	15
3.2	Schematic diagram of compact tension specimen (CT specimen)	16
3.3	New model preference for the specimen analysis	17
3.4	Geometry preferences for the specimen analysis	18
3.5	Points of the specimen drawing	19
3.6	Curve of the specimen drawing	19
3.7	Created surface for the specimen drawing	20
3.8	Created curves before breaking the surface into parts	20
3.9	Broken surfaces after curves deletion	21
3.10	Mesh seed for L2 = 0.2.	22
3.11	Mesh shape settings	23
3.12	Created mesh on the specimen	23
3.13	Specimen with boundary layers	24
3.14	Time series generator network	25
3.15	Time Series Generator properties window	26
3.16	Generated time series interface	27
3.17	ASCII Translate setting interface	28
3.18	Strain-life analysis network	29
3.19	Strain-life properties setting	30
3.20	Material Map for the strain-life analysis	31
3.21	Strain-life result display	32

4.1	Microstrain Value for Dynamic Failure Analysis for Compression-Compression Case at 1 Hz	35
4.2	Damage Contour for Dynamic Failure Analysis for Compression-Compression Case at 1 Hz	35
4.3	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Compression Case at 1 Hz	36
4.4	Microstrain Value for Dynamic Failure Analysis for Compression-Compression Case at 2 Hz	37
4.5	Damage Contour for Dynamic Failure Analysis for Compression-Compression Case at 2 Hz	37
4.6	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Compression Case at 2 Hz	38
4.7	Microstrain Value for Dynamic Failure Analysis for Compression-Compression Case at 3 Hz	39
4.8	Damage Contour for Dynamic Failure Analysis for Compression-Compression Case at 3 Hz	39
4.9	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Compression Case at 3 Hz	40
4.10	Microstrain Value for Dynamic Failure Analysis for Compression-Tension Case at 1 Hz	41
4.11	Damage Contour for Dynamic Failure Analysis for Compression-Tension Case at 1 Hz	41
4.12	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Tension Case at 1 Hz	42
4.13	Microstrain Value for Dynamic Failure Analysis for Compression-Tension Case at 2 Hz	43
4.14	Damage Contour for Dynamic Failure Analysis for Compression-Tension Case at 2 Hz	43
4.15	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Tension Case at 2 Hz	44
4.16	Microstrain Value for Dynamic Failure Analysis for Compression-Tension Case at 3 Hz	45

4.17	Damage Contour for Dynamic Failure Analysis for Compression-Tension Case at 3 Hz	45
4.18	Damage Contour at Notch Point for Dynamic Failure Analysis at for Compression-Tension Case at 3 Hz	46
4.19	Microstrain Value for Dynamic Failure Analysis for Tension-Tension Case at 1 Hz	47
4.20	Damage Contour for Dynamic Failure Analysis for Tension-Tension Case at 1 Hz	47
4.21	Damage Contour at Notch Point for Dynamic Failure Analysis at for Tension-Tension Case at 1 Hz	48
4.22	Microstrain Value for Dynamic Failure Analysis for Tension-Tension Case at 2 Hz	49
4.23	Damage Contour for Dynamic Failure Analysis for Tension-Tension Case at 2 Hz	49
4.24	Damage Contour at Notch Point for Dynamic Failure Analysis at for Tension-Tension Case at 2 Hz	50
4.25	Microstrain Value for Dynamic Failure Analysis for Tension-Tension Case at 3 Hz	51
4.26	Damage Contour for Dynamic Failure Analysis for Tension-Tension Case at 3 Hz	51
4.27	Damage Contour at Notch Point for Dynamic Failure Analysis at for Tension-Tension Case at 3 Hz	52
4.28	Microstrain Value for Static Failure Analysis for Compression-Compression Case at 1 Hz	53
4.29	Damage Contour for Static Failure Analysis for Compression-Compression Case at 1 Hz	53
4.30	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Compression Case at 1 Hz	54
4.31	Microstrain Value for Static Failure Analysis for Compression-Compression Case at 2 Hz	55
4.32	Damage Contour for Static Failure Analysis for Compression-Compression Case at 2 Hz	55

4.33	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Compression Case at 2 Hz	56
4.34	Microstrain Value for Static Failure Analysis for Compression-Compression Case at 3 Hz	57
4.35	Damage Contour for Static Failure Analysis for Compression-Compression Case at 3 Hz	57
4.36	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Compression Case at 3 Hz	58
4.37	Microstrain Value for Static Failure Analysis for Compression-Tension Case at 1 Hz	59
4.38	Damage Contour for Static Failure Analysis for Compression-Tension Case at 1 Hz	59
4.39	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Tension Case at 1 Hz	60
4.40	Microstrain Value for Static Failure Analysis for Compression-Tension Case at 2 Hz	61
4.41	Damage Contour for Static Failure Analysis for Compression-Tension Case at 2 Hz	61
4.42	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Tension Case at 2 Hz	62
4.43	Microstrain Value for Static Failure Analysis for Compression-Tension Case at 3 Hz	63
4.44	Damage Contour for Static Failure Analysis for Compression-Tension Case at 3 Hz	63
4.45	Damage Contour at Notch Point for Static Failure Analysis at for Compression-Tension Case at 3 Hz	64
4.46	Microstrain Value for Static Failure Analysis for Tension-Tension Case at 1 Hz	65
4.47	Damage Contour for Static Failure Analysis for Tension-Tension Case at 1 Hz	65
4.48	Damage Contour at Notch Point for Static Failure Analysis at for Tension-Tension Case at 1 Hz	66

4.49	Microstrain Value for Static Failure Analysis for Tension-Tension Case at 2 Hz	67
4.50	Damage Contour for Static Failure Analysis for Tension-Tension Case at 2 Hz	67
4.51	Damage Contour at Notch Point for Static Failure Analysis at for Tension-Tension Case at 2 Hz	68
4.52	Microstrain Value for Static Failure Analysis for Tension-Tension Case at 3 Hz	69
4.53	Damage Contour for Static Failure Analysis for Tension-Tension Case at 3 Hz	69
4.54	Damage Contour at Notch Point for Static Failure Analysis at for Tension-Tension Case at 3 Hz	70

LIST OF SYMBOLS

D	Fatigue damage
E	Modulus of Elasticy
ε	Strain
f	Frequency
K_I	Stress intensity factor
N_f	Number of cycle to failure for a particular stress range and mean
N_i	Number of cycle to failure within a particular stress range and mean
v	Poisson's ratio
P	Load
σ	Theoretical failure stress
$\sigma_F^{'}$	Fatigue strength coefficient
S_{ut}	Ultimate Tensile Strength
W	Width
$\frac{L2}{L1}$	Mesh size ratio

LIST OF ABBREVIATIONS

AISI	American Iron and Steel Institute
CT	Compact tension
DCB	Double cantilever beam
FEA	Finite Element Analysis
SAE	Society of Automotive Engineers
SENB	Single edge notched bend
US	The United State

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This project concerns about the parametric study on fatigue failure using crack initiation method. The effect of microstrain values and the frequency that applied to the specimen were investigated using finite element software. The analysis is applied on AISI/SAE 1045 steel (the following stated as AISI 1045 steel to refer AISI/SAE 1045 steel), which is widely used in the automotive applications. AISI 1045 steel was chosen for the project since that this material plays important roles in the engineering field. AISI 1045 steel is usually used as axles, bolts, connecting rods, studs, crankshafts, torsion bars, sockets, etc. So, this material has selected simply because of the importance of AISI 1045 steel in daily life applications, which are require high safety measures. Due to its wide use, there are also huge possibilities that it will be failed in their applications. Some of the failures are because of they become fatigue, due to the repeating force acted on the material and cause them unable to work perfectly and then fail.

Fatigue failure is a common type of failure which is hardly to determine their life span. Basically, this type of failure is started when a crack tip has appeared under certain circumstances which also known the crack initiation, allowing the crack to propagate, making the material become weaker by times. The crack propagation is commonly happened by the cyclic loads or forces that acted on the structure, which applied continuously along with time passes, though the forces and loads did not ever reach the ultimate strength of the material.

In order to estimate the life span of the metal structures, some experiments and analyses must be conducted. This is important to determine the limits of the materials before they are selected to be used in the design. The works on predicting the life cycle of AISI 1045 steel is started by determining the stress distribution on the material by conducting the linear static analysis. The analysis is done using standard compact tension (CT) specimen as the model testing, using MSC Patran 2010 and MSC Nastran 2007 softwares. The results of the analysis is then will be used as the reference to continue the study on parametric studies on fatigue failure of AISI 1045 steel using crack initiation method.

1.2 PROBLEM STATEMENT

Since the industrial revolution there are lots of new machines were created to meet human needs in their daily activities and also the need of the development of the industrial activities. Since then, there are also lots of new materials used for the latest machines. The usages of the new machines bring the daily activities alive with the times.

However in the past days, the creations of the new machines were not considering one of the material weaknesses in their application which bring them into failure. The failure of the materials did a lot of damage and lost while some of the failures were also bring fatality to humans.

Basically, fatigue of materials occurs as they were used repeatedly subjected to reverse stresses over the time. When the materials reach its maximum cycles, it no longer can hold the load as it did before. Due to fatigue, the material fails and bring catastrophe to the mankind and also to the surroundings. Some of the catastrophes that involved fatality due to fatigue were two 1954 Comet jet planes were crashed due to metal fatigue. The first crash killed 29 passengers and a crew lost their lives. Three months later, the second crash killed fourteen passengers and seven crews (British Broadcasting Corporation). However, there was 34 years later, another airline catastrophe really did change the airline industry related to the material control. It was

1988 Aloha Flight 243 that a large section of the fuselage blown off, leaving dozens of passengers rode the plane in open-air breeze (Nolan, 2007).

Since then a lot of actions were taken to prevent any other catastrophe that happens due to fatigue. Huge number of engineers made the research on metal fatigue due to dynamic loading. Some of the tests were also involved the study of crack initiation which is then lead the further stage of fatigue failure.

1.3 PROJECT OBJECTIVES

There are several objectives to be achieved in this study;

(i) To investigate the fatigue failure of AISI 1045 steel by varying the microstrain parameters under different loading cases.

(ii) To predict the crack initiation life of AISI 1045 steel by using crack initiation method.

1.4 PROJECT SCOPES

This project is focusing on the parametric studies on fatigue failure using crack initiation method. This focus area is done based on the following aspects:

(i) AISI 1045 steel is used as the material that to be studied.

(ii) The crack initiation method by using damage equation is used to predict the initial life cycle of the material.

(iii) Compact Tension Specimen (CT) is used as the model of the test.

(iv) Linear Static Stress Analysis is conducted using MSC Patran 2010 and MSC Nastran 2007 software.

(v) Fatigue Analysis is carried out using ANSYS 13.0 nCode DesignLife software.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The chronology of this chapter has been properly organized according to the sequence of this study. In this chapter, the readers will get chances to understand the idea on the importance of failure analysis in the engineering field. The brief of AISI/SAE 1045 steel in section (2.2) will give some information about steels, the properties and classifications. The main part of this section is briefing about AISI/SAE 1045 steel, which will be used for this project. Starting from section (2.3), readers will be given with a short overview about this project, by introducing the metal failure analysis including its analysis types. In the same section, the readers also will be introduced with an overview about finite element analysis (FEA) including the steps of FEA. Tough the literature review about FEA is likely quite general, however this project uses MSC Patran and MSC Nastran as default FEA software for static linear analysis and ANSYS 13.0 nCode DesignLife for dynamic analysis; and in order to set up the model and run the analysis as well as obtaining the analysis result of the project. Section (2.4) is then will introduce about crack initiation with briefs about fatigue and crack initiation life. This section will bring the readers to the relations of fatigue damage value and crack initiation life cycles in order to precede this project into the analysis. The following sections will show the brief of the specimen geometry (2.7) will give an introduction about CT specimens while exposing some other specimens that exist for related kind of analysis which are likely almost or same to this project. However, CT specimen is the only specimen that will be discussed more than the others since that this project only use CT specimen in the analysis job to obtain the prediction of crack initiation life cycle of AISI/SAE 1045 steel using CT specimen. The last section will give the summary of this chapter in proceeding to the next chapter of this project.

2.2 AISI/SAE 1045 STEEL

2.2.1 Steel

Metals and alloys are widely applied in engineering designs as they have many engineering properties. Some of the most used metal type in the engineering design is iron and its alloys which are also known as steel. It has accounted for huge production of metals and becoming the most favourite metal chosen by the engineers mainly because of the combination of good strength, toughness, and ductility at low cost. In smaller portions of steel types, there are variety of steel family such as plain-carbon steels, alloy steels, stainless steels, cast iron and copper alloys. They are widely used in manufacturing of various types of parts and tools, which are having the needs of high rotation and force acted on them. In other classification, steel is categorised as ferrous alloys, since it is based on iron alloys (Smith & Hashemi, 2006).

2.2.2 Carbon Steels Classifications

The definition of The American Iron and Steel Institute (AISI) of carbon steel is; steel is considered to be carbon steel when no minimum content is specified or required for chromium, cobalt, columbium [niobium], molybdenum, nickel, titanium, tungsten, vanadium or zirconium, or any other element to be added to obtain a desired alloying effect; when the specified minimum for copper does not exceed 0.40 per cent; or when the maximum content specified for any of the following elements does not exceed the percentages noted: manganese 1.65, silicon 0.60, copper 0.60 (Key to Metals, 2011).

Carbon steel can be classified according to various deoxidation practices, which have effects on the steel properties. Variations of the carbon content in the steels also give a huge effect on the mechanical properties; with increasing the carbon content will lead to the increase the strength and hardness of the steel. Generally, carbon steels are divided into groups, according to their carbon content, which the carbon steels that contain up to 2% total alloying elements and can be subdivided into low carbon steels, medium carbon steels, high carbon steels, and ultrahigh carbon steels.

The most widely used identification systems for carbon and alloy steels in the United State (US) is AISI and Society of Automotive Engineers (SAE), which are usually used four-digit identifications. The first digit of the identification is used to indicate the grouping by major alloying element, while the second digit in some instances suggests the relative percentage of the primary alloying element in the series. The remaining digits indicate the median carbon content in hundredths of a percent (Budinski & Budinski, 2010).

2.2.3 AISI/SAE 1045 Steel

AISI 1045 steel is designated as high carbon steel where the percentage of carbon in the elemental composition is higher than 0.3%. High carbon steel is one of the types of steel with more than 0.3% carbon. It is a tough and hard type of steel, due to its high amount of carbon content; however make it less formal steel compared to low carbon steel and medium carbon steel. High carbon steel's hardness makes it suitable for plow blades, shovels, bedsprings, cutting edges, or other high-wear applications (American Iron and Steel Institute). The elemental composition of AISI 1045 steel is shown in the Table 2.1.

Table 2.1: AISI/SAE 1045 steel element composition.

Element Composition (mass fraction, in %)	
Parameter	**Value**
Aluminium (total)	0.040
Carbon	0.462
Chromium	0.036

Source: National Institute of Standard and Technology (2012)

AISI 1045 steel is usually used in automotive industries in making car parts. Some of the parts are including axles, bolts, connecting rods, studs, crankshafts, torsion bars, light gears, guide rods, etc. Thus, the applications of AISI 1045 steel acquire a lot of vibrations and cyclic loadings acted on the parts.

The usage of the AISI 1045 steel is considered by its mechanical properties which are usually hard and tough for high wear and tear. The mechanical properties of AISI 1045 steel are shown in Table 2.2.

Table 2.2: AISI/SAE 1045 steel mechanical properties.

Grade AISI/SAE	Hardness HB	Tensile Strength S_{ut} MPa	Modulus of Elasticity E GPa	Fatigue Strength Coefficient σ'_F MPa
1045	225	725	200	1225
1045	410	1450	200	1860
1045	390	1345	205	1585
1045	450	1585	205	1795
1045	500	1825	205	2275
1045	595	2240	205	2725

Source: Budynas & Nisbett (2010)

2.3 INTRODUCTION OF METAL FATIGUE FAILURE ANALYSIS

Since the growth of the technology of metallurgy engineering and the expanding of engineering knowledge based on the success of the high technology appliance in the human activities, there are so many latest inventions and innovations are using metal as the final products that meet the consumers. However, the important part of using metal is the prediction on the capability of the metal to work until it reaches the failure stage because of certain conditions that acted on the metal. Thus the study on predicting the working capability limit has known as the 'failure analysis.'

2.3.1 Types of Fatigue Failure Analysis

Failure Analysis is a method which is gathering the data and doing the analysis to determine the cause of the failure. In general speaking, the metal failures could be attributed by varying types of cases or the combination of two or more cases such as the improper material selection, design, maintenance, production defects, weather, workloads and the exposure of the metal to the working surroundings (Das, 1997).

The analysis might be done by doing the experiment on the materials by testing under certain conditions and sample types. Some of the failure analysis is done by testing the metals under static forces to get the ultimate strength of the material by applying the forces to the material until it fails. This type of failure analysis is usually done to determine the maximum load that can be applied on the material when it is used in the actual scenario. Besides, there is also another type of analysis which is newer in modern engineering which is dynamic testing. In a simple meaning, dynamic testing on the material is done to determine the maximum life of the material under certain circumstances which are might be or might not be related to the maximum load that can be applied on the material.

In another way, failure analysis also can be conducted by doing the simulated testing by using any engineering analysis software. In modern engineering, this method is likely more practical since that this method can optimize the source by reducing the cost, energy, time, and wasted sources. This kind of testing is also known as 'Finite Element Analysis (FEA)' under the method name of 'Finite Element Method (FEM).'

2.3.2 Finite Element Analysis (FEA)

FEA is a method that using the numerical methods to simulate and analyse the materials by finding the approximate solutions to the field problems (Widas, 1997). Though that FEA can be done by using manual calculation, however the development of modern engineering has made the analysis easier since that there is a lot of analysis software that produced to meet the engineering needs nowadays. The birth of the latest version of FEA software gives better results on the analysis and capable to analyse very complex problems. The latest FEA software is also capable to analyse for static, dynamic, heat and vibration analysis.

Basically, FEA is consisting of three major steps, pre-processing, analyser, and post-processing; which are the important parts in the analysis jobs by using the FEA.

2.3.2.1 Pre-processing.

Pre-processing step is the stage where the user constructs a model of a part that to be analysed in which the geometry is divided into a number of discrete sub-regions, which is also called as ‘nodes’. The nodes are generated by building the mesh on the model. Some of the pre-processors are also can overlay mesh by importing pre-existing CAD files, besides of building the model itself in the pre-processors. When the nodes are ready, boundary conditions are then applied on the model along with the material types and properties (Roylance, 2011).

2.3.2.2 Analysis

Analysis which is also known as solver is where the datasets prepared by the pre-processor is used as input to generate and reassemble acquired equations into the analysis to obtain the result (Roensch, 2008); by constructing and solving the linear and non-linear algebraic equation;

$$K_{ij}u_j = f_i \tag{2.1}$$

Where *u* and *F* are the displacements and externally applied forces at the nodal points and *K* is the system stiffness matrix. Matrix *K* is varies depending on the type of problem that is to be analysed. The analysis job is done by the software and the final results of the analysis are depending on the setups during the pre-processing.

2.3.2.3 Post-processing

Post-processing is the last step of the FEA analysis which is the process of the investigation of the results after pre-processing and solving processes. The results obtained from the analysis will be assisted with graphical and managed result data. During the end of this period, the final conclusions or predictions of the study might be addressed depending on the pre-processing setup and analysis jobs result.

2.4 CRACK INITIATION

2.4.1 Fatigue

Fatigue mechanism knowledge is an essential in order to consider various technical conditions which will affect the specimen fatigue life and fatigue crack growth, such as material surface quality, residual stress and environmental influence. It is also important for the analysis of fatigue properties of an engineering structure and designs.

The fatigue life is usually split to a crack initiation period and a crack growth period. The initiation period usually including the microcrack growth, but it is too small to be detected by naked eyes. The next period of the fatigue life, the crack is growing until complete failure, which is also known as crack propagation. The crack propagation started because of the uneven stress distribution on a micro level, with the stress concentration at the crack tip. It then makes the neighbouring grains becoming weaker by times and another slip system has been activated (Schijve, 2001).

2.4.2 Crack Initiation Life

Crack initiation life is usually predicted by using local stress-strain approach, which is the most widely accepted approach to predict the crack initiation life. By postulating the number of the cycles required to initiate a crack in the initial zone, N_i to be equal to the number of cycles to produce failure of the specimen in a laboratory test under the same cyclic strains and stresses, this concept has been illustrated as the notch at the critical zone, which indicate as the initiation of the crack (Collins, 2003).

In the calculation of crack initiation life, the value of fatigue damage, D is used in order to obtain the number of cycles to initiate the crack. The equation of the fatigue damage value is obtained from Palmgren-Milner rule, for the accumulated damage, ΣD is expressed as the equation below, where N_i is the number of cycles within a particular stress range and mean while N_f is the number of cycles to failure for a particular stress range and mean (Abdullah, *et. al*, 2011).

$$\sum D = \sum \frac{N_i}{N_f} \tag{2.2}$$

The equation is then extracted with the fatigue damage value for each cycle, and it can be obtained by the equation below.

$$D = \frac{1}{N_f} \tag{2.2}$$

From the damage values that obtained from every cycle, they are then can be determine the value of the crack initiation cycle by obtaining the value of N_f.

2.5 SPECIMEN GEOMETRY

2.5.1 Standard Test Specimens

Standard test specimens are used for experimental application to determine fatigue life of materials. The specimens that are selected depend on the type of fatigue test, since those not all standard specimens are suitable for any type of tests. The types are included compact tension specimen (CT), single edge notched bend specimen (SENB), double cantilever beam specimen (DCB) and others. CT specimen usually used for fatigue crack analysis, which is a notched sample. Notched sample is used to create fatigue crack by cycling the sample to the maximum and the minimum loads. When it comes the time where the fatigue crack started to appear at the notch, it will expand trough the sample. It is a good representation of the real situation of fatigue crack scenario.

2.5.2 Compact Tension Specimen

As proposed by William H. Hartt (Hartt, 1991) in his on assesses and characterizes the fatigue properties of several new high strength structural steels under